A Numerical Simulation in Region with Very Large Aspect Ratio Using Some Sub Domain Grids

Anna KUWANA, Kanako IKEDA and Tetuya KAWAMURA

(Received December 10, 2012)

Abstract
A new numerical method that is suitable for the calculation of incompressible flow in a pipe of very large aspect ratio is developed. The idea of the method is that the original flow is expressed as sum of the "main flow" and the variation from the main flow. We applied the new method for incompressible 2-dimensional pulsatile flows in a long pipe and obtained good results in the previous studies. In this study, the new method is extended for 3-dimensional flows with heat and thermal convection in some orthogonally bended pipes using some sub domain grids since such thermal convections in a very long region are important for disaster prevention or manufacturing.

1 Introduction

We can find a lot of incompressible fluid flows in very long regions such as rivers, blood vessels, tunnels and pipes. The main problem to calculate flows in long regions is that it is quite difficult to satisfy the equation of continuity precisely. If we use the method based on the stream-function and vorticity ($\psi - \omega$ method), we are free from the problem mentioned above. However this method works only on the two dimensional or axi-symmetric flows. Therefore the MAC method [1] and its variations [2] are preferred to treat 3-dimensional flows and the flows in regions of complicated shapes. On the other hand, the conservation of mass (i.e. the equation of continuity) is not satisfied very well by the method based on the Poisson equation of pressure appearing in the MAC method. It is also difficult to solve the Poisson equation precisely by using the iterative method. This disadvantage is emphasized in the case of the region of large aspect ratio (length / diameter).

In the previous studies we presented a new numerical method that is suitable for the calculation of incompressible fluid flow in a very long region [3, 4]. In the method, the original flow is expressed as sum of the "main flow" and the variation from the main flow. The former is obtained by solving 1-dimensional Navier-Stokes equation analytically.

This method is extended for flows with heat and thermal convection in a vertical pipe on the assumption that fire disaster in tunnels, metro stations or elevator shafts [5, 6]. In this study, 3-dimensional flows with heat and thermal convection in some orthogonally bended pipes are calculated using some sub domain grids.

2 Numerical Method

The computational region is a long pipe standing vertically. The heat source locates on the wall as shown in Fig.1 (note that figures of long pipes are rotated 90 degrees to save the space in this paper). The convective flow is induced by this heat source.

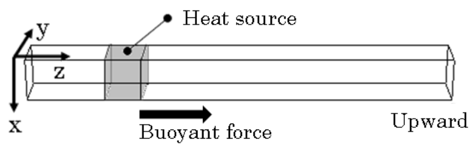


Fig. 1: Computational domain and location of heat source.

The basic equation for incompressible 3-dimentional flow under the Boussinesq approximation is as follows:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z} = 0 \tag{1}$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z} = -\frac{\partial p}{\partial x}+\frac{1}{\mathrm{Re}}(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}) \tag{2}$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z} = -\frac{\partial p}{\partial y}+\frac{1}{\mathrm{Re}}(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}) \tag{3}$$

$$\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z} = -\frac{\partial p}{\partial z}+\frac{1}{\mathrm{Re}}(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2})+\frac{\mathrm{Gr}}{\mathrm{Re}^2}T \tag{4}$$

$$\frac{\partial T}{\partial t}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z} = \frac{1}{\mathrm{Re}\cdot\mathrm{Pr}}(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}) \tag{5}$$

where (x, y, z) is the spatial coordinate (z means vertical direction as is shown in Fig.1), (u, v, w) is the velocity, T is the temperature, p is the pressure, and parameters Re, Gr and Pr are Reynolds, Grashof and Prandtl numbers respectively. Reynolds number is set to 50 and Grashof number is set to 5000 so as to $\mathrm{Gr}/\mathrm{Re}^2 = 2$. These values are determined by considering both the effect of turbulent eddy viscosity and the generation of upward flow within a short period. Prandtl number is set to 0.71 assuming air.

A conceptual diagram of this method is shown in Fig.2. The original flow (continuous arrows) is expressed as sum of the "main flow (dashed arrows)" and the variation from the main flow (dotted arrows). The "main flow" satisfies the equation of continuity approximately.

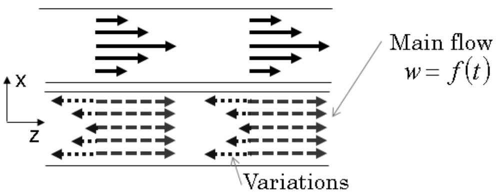


Fig. 2: A conceptual diagram of proposed method.

If the computational region is very long in vertical direction (z direction), the vertical flow (w) is dominant and can be treated as nearly 1-dimensional. In other words, We can assume

$$w = w(z,t), \quad p = p(z,t), \quad u = v = 0, \quad \frac{\partial}{\partial x} = \frac{\partial}{\partial y} = 0 \tag{6}$$

as the first approximation. Substituting equation (6) into equations (1), (4) with $T = 0$, we obtain

$$\frac{\partial w}{\partial z} = 0 \tag{7}$$

$$\frac{\partial w}{\partial t} = -\frac{\partial p}{\partial z} \tag{8}$$

(Equations (2), (3) and (5) become $0 = 0$) We also neglect the buoyancy force, because we need a simple one-dimensional flow here. From equation (7) and the first one of equation (6), we obtain

$$w = f(t) \tag{9}$$

as a "main flow" or "dominant flow". In our previous study, $f(t)$ is determined from boundary condition at the entrance of the long region (pulsatile inflow). In this case, there is no main flow that approximates the original flow in the first stage of calculation. So, we determine it from the averaged velocity in the region surrounded by the heat source [6].

If we substitute equation (9) into (8), we obtain

$$f'(t) = -\frac{\partial p}{\partial z} \quad i.e. \quad p = -f'(t)z + C(t) \quad (C(t) : Arbitrary function) \tag{10}$$

From equations (9), (10), we can express the original flow as sum of the "main flow" and the "variation of the velocity and pressure $(\tilde{w}, \tilde{p})$" as follows:

$$w = f(t) + \tilde{w}(x, y, z, t), \quad p = -f'(t)z + C(t) + \tilde{p}(x, y, z, t) \tag{11}$$

After the substitution of equation (11) into the original equations (1)-(5), we obtain the basic equation in this study as follows:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial \tilde{w}}{\partial z} = 0 \tag{12}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + (f + \tilde{w})\frac{\partial u}{\partial z} = -\frac{\partial \tilde{p}}{\partial x} + \frac{1}{\mathrm{Re}}(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}) \tag{13}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + (f + \tilde{w})\frac{\partial v}{\partial z} = -\frac{\partial \tilde{p}}{\partial y} + \frac{1}{\mathrm{Re}}(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}) \tag{14}$$

$$\frac{\partial \tilde{w}}{\partial t} + u\frac{\partial \tilde{w}}{\partial x} + v\frac{\partial \tilde{w}}{\partial y} + (f + \tilde{w})\frac{\partial \tilde{w}}{\partial z} = -\frac{\partial \tilde{p}}{\partial z} + \frac{1}{\mathrm{Re}}(\frac{\partial^2 \tilde{w}}{\partial x^2} + \frac{\partial^2 \tilde{w}}{\partial y^2} + \frac{\partial^2 \tilde{w}}{\partial z^2}) + \frac{\mathrm{Gr}}{\mathrm{Re}^2}T \tag{15}$$

$$\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + (f + \tilde{w})\frac{\partial T}{\partial z} = \frac{1}{\mathrm{Re}\cdot\mathrm{Pr}}(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}) \tag{16}$$

These equations are nearly the same as the original equations (1)-(5) although unknowns are different. These equations can be solved by the standard method such as the MAC method [1], SMAC method [2] and so on. Note that the boundary condition on the wall (no-slip: $w = 0$) becomes $\tilde{w} = -f(t)$.

Thermal convection in three orthogonally bended pipe shown in Fig.3 is calculated using three sub domain grids in this study. We call them sub domains A, B, C respectively.

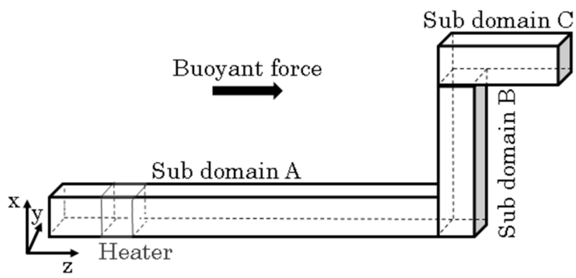


Fig. 3: Orthogonally bended pipes expressed by three sub domains.

The "main flow" of sub domains A and C is parallel to z-axis, the same direction of gravity, so that equations (11) and (12)-(16) can be used. In sub domain B, since the "main flow" is parallel to x-axis, equations (17) and (18)-(22) can be used instead of them.

$$u = f(t) + \tilde{u}(x, y, z, t), \quad p = -f'(t)x + C(t) + \tilde{p}(x, y, z, t) \tag{17}$$

$$\frac{\partial \tilde{u}}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{18}$$

$$\frac{\partial \tilde{u}}{\partial t} + (f + \tilde{u})\frac{\partial \tilde{u}}{\partial x} + v\frac{\partial \tilde{u}}{\partial y} + w\frac{\partial \tilde{u}}{\partial z} = -\frac{\partial \tilde{p}}{\partial x} + \frac{1}{\mathrm{Re}}(\frac{\partial^2 \tilde{u}}{\partial x^2} + \frac{\partial^2 \tilde{u}}{\partial y^2} + \frac{\partial^2 \tilde{u}}{\partial z^2}) \tag{19}$$

$$\frac{\partial v}{\partial t} + (f + \tilde{u})\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = -\frac{\partial \tilde{p}}{\partial y} + \frac{1}{\mathrm{Re}}(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}) \tag{20}$$

$$\frac{\partial w}{\partial t} + (f + \tilde{u})\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} = -\frac{\partial \tilde{p}}{\partial z} + \frac{1}{\mathrm{Re}}(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}) + \frac{\mathrm{Gr}}{\mathrm{Re}^2}T \tag{21}$$

$$\frac{\partial T}{\partial t} + (f + \tilde{u})\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z} = \frac{1}{\mathrm{Re}\cdot\mathrm{Pr}}(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}) \tag{22}$$

If cross sectional areas of sub domains A, B and C are different, the "main flows" in sub domains are adjusted so that flow rates through the pipe at all cross sections in three sub domains become the same value. For example, if cross sectional areas of sub domains A, B, C are 2: 1: 3, the "main flows" in sub domains A, B, C are $\frac{1}{2}f(t) : f(t) : \frac{1}{3}f(t)$ respectively.

The physical quantities are calculated with respect to each sub domain independently. The quantities on the boundary at the joint regions are given as boundary condition for each sub domain on every time step [7]. The quantities are calculated by equation (23), i.e. arithmetic weighted mean between the joint regions as shown in Fig.5.

$$\frac{dx_B}{dx_A + dx_B}U_A + \frac{dx_A}{dx_A + dx_B}U_B \tag{23}$$

When equation (23) is calculated, we use the physical quantity of the "original flow" by equations (11) and (17) as U_A and U_B, not the "variations" calculated by equations (12)-(16) and (18)-(22).

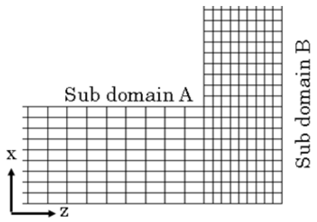


Fig. 4: Grid points near the joint region.

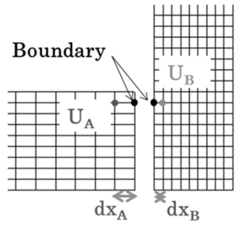


Fig. 5: Connection of sub domains.

3 Results

3.1 Pulsatile Inflow

First, calculations are carried out in the pipe with pulsatile inflow where good results were obtained in the previous studies. Temperature is set to zero at all grid points in the region. Pulsatile flow [1] is assumed to come in from the entrance of the sub domain A. The exit of the sub domain C is free outflow.

Figure 7 shows time history of variation of the flux [2] of the pipe at the entrance of the pipe and the center of each sub domain. The position of measurement points are shown in Fig.6. The graph of flux versus time, Fig.7 (a), shows that fluxes at all measurement points are the same. Compared to the result of Fig.7 (b) by conventional MAC method whose computational cost is in the same range with proposed method of Fig.7 (a), flux decreases with distance from the entrance of the pipe.

Flux calculated by the conventional method can be kept if the conservation of mass is satisfied very well by the method based on the Poisson equation of pressure, of course. The result obtained by the conventional MAC method with large iterative times is shown in Fig.7 (c).

Figures 8 and 9 show velocity vector at the cross section in center of y-direction. Pulsatile flow comes in from the entrance of the sub domain A (left-hand side of Fig.8). Flux (i.e. the magnitude of vector, in these figures) is kept from the entrance to the exit by the results of proposed method.

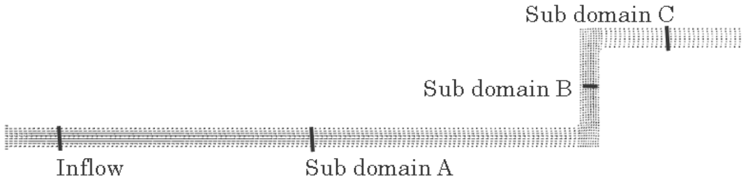
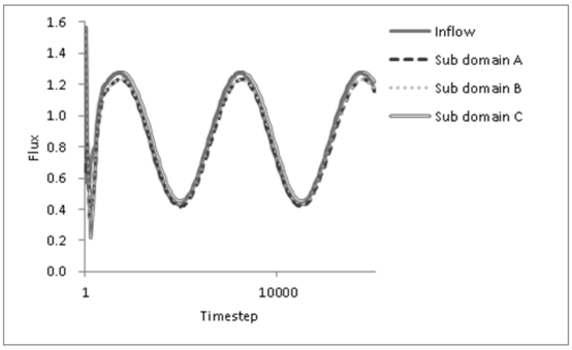
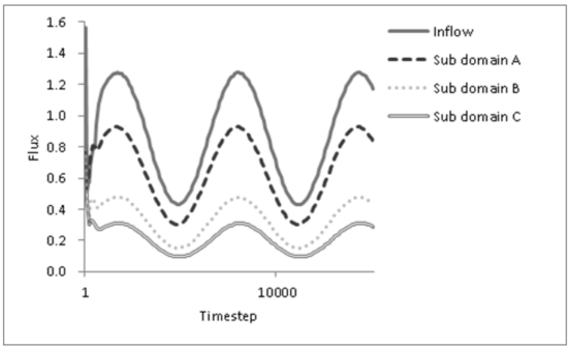


Fig. 6: Cross sections for calculation of flux.

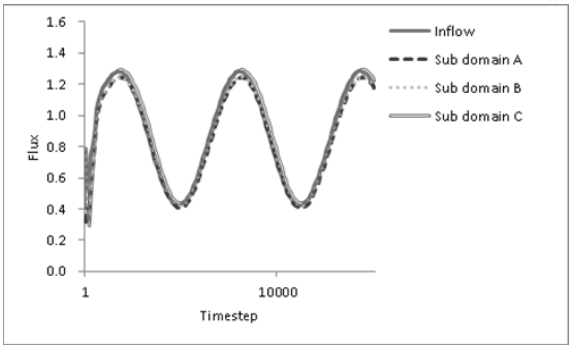
[1]Pulsatile flow: $w = a + bsin(c \times dt)$, a, b, c: reasonable constants.
[2]Flux: $\sum w \times dxdy$ in the sub domains A and C, $\sum u \times dydz$ in the sub domain B.



(a) Proposed method.

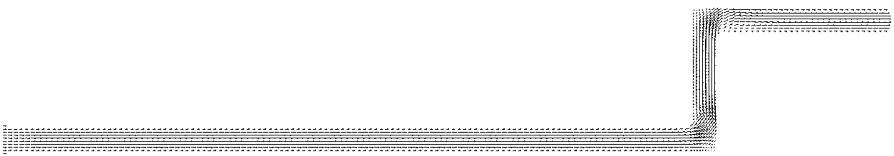


(b) Conventional method,
same computational cost with (a).

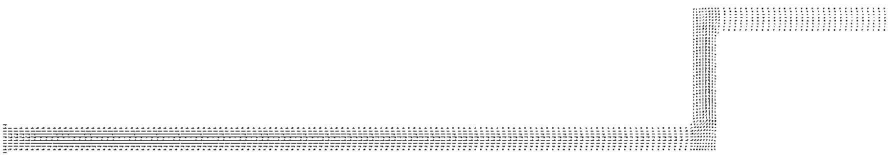


(c) Conventional method.

Fig. 7: Time history of variation of the flux of the pipe.

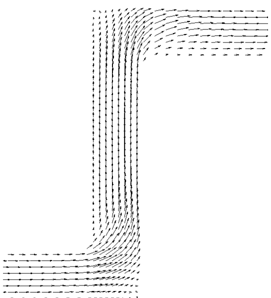


(a) Proposed method.

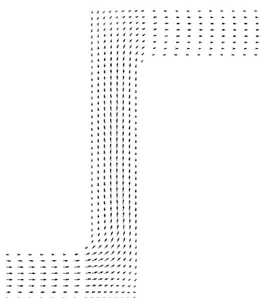


(b) Conventional method, same computational cost with (a).

Fig. 8: Velocity vector of the pipe.



(a) Proposed method.



(b) Conventional method,
same computational cost with (a).

Fig. 9: Velocity vector of the pipe near the joint region.

3.2 Thermal Convection

Next, thermal convection by heater near the entrance is calculated without pulsatile inflow. The wall except the heater is assumed to be adiabatic, initial condition of temperature and velocity is zero, the entrance and exit of the pipe is free inflow and outflow.

Figure 10 shows the position of the heater and measurement points for flux. Figure 11 is the graph of flux versus time. Figures 12, 13 show temperature field and velocity vector at the cross section in the center of y-direction respectively.

As shown in these figures, movement toward the exit flow increases gradually, and heated flow from the heater reaches the exit of the pipe at timestep ≈ 5000, after that, the flow reaches nearly steady state. At this time, fluxes at all measurement points are the same.

Compared to the result obtained by conventional MAC method whose computational cost is in the same range with proposed method, the flow seems strange since it takes a while to reach the exit of the pipe (Timestep ≈ 25000), after that the flow is accelerates unnaturally with distance from the entrance of the pipe.

Flux calculated by the conventional method can be kept if the conservation of mass is satisfied very well, of course.

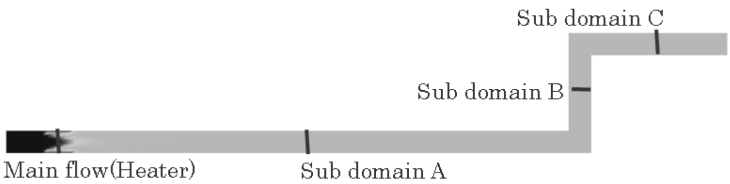
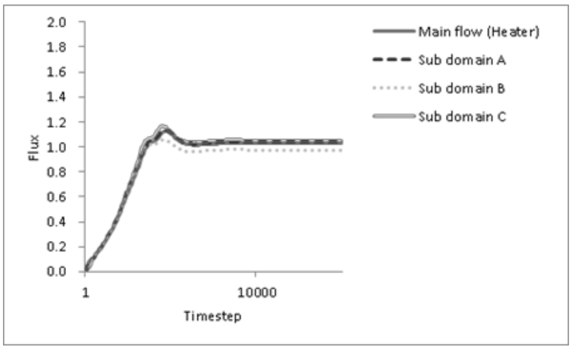
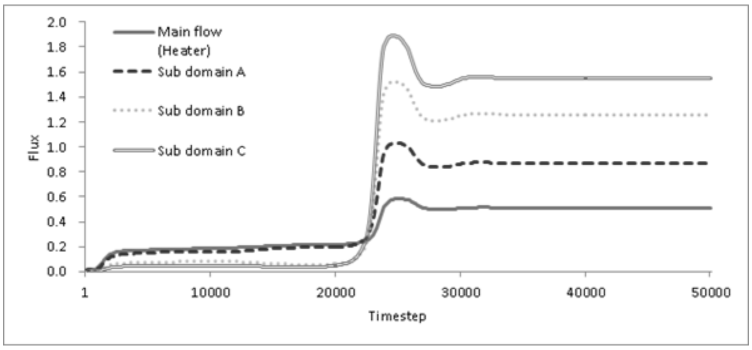


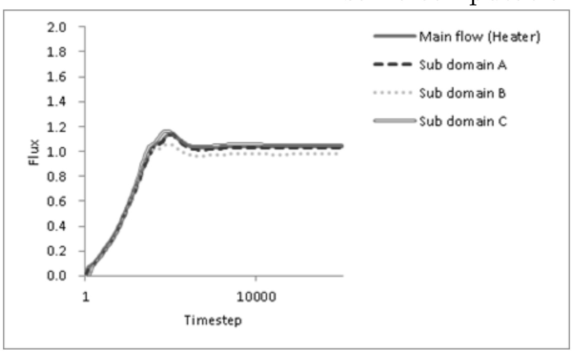
Fig. 10: Cross sections for calculation of flux.



(a) Proposed method.

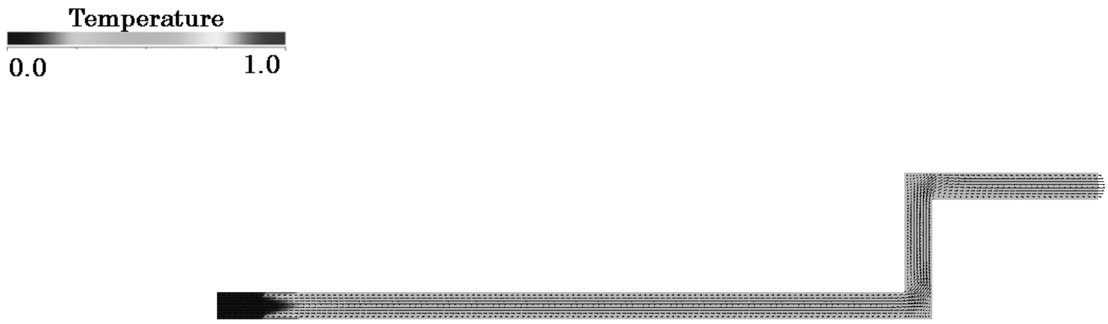


(b) Conventional method, same computational cost with (a).

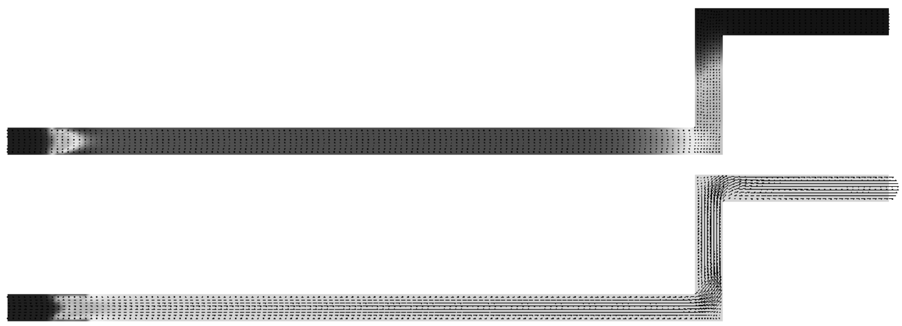


(c) Conventional method.

Fig. 11: Time history of variation of the flux of the pipe.

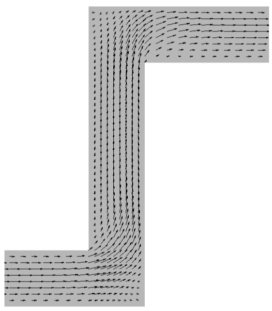


(a) Proposed method (Timestep=15000).

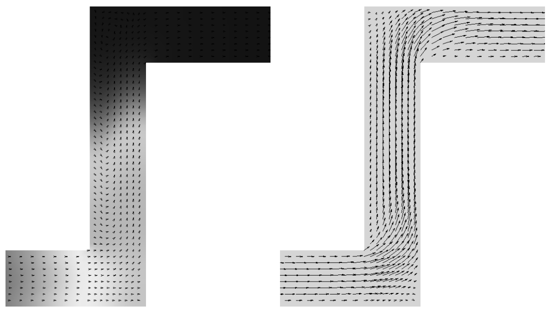


(b) Conventional method, same computational cost with (a) (Timestep=15000, 50000).

Fig. 12: Temperature and velocity vector of the pipe.



(a) Proposed method (Timestep=15000).



(b) Conventional method, same computational cost with (a) (Timestep=15000, 50000).

Fig. 13: Temperature and velocity vector near the joint region.

3.3 Thermal Convection with Different Cross Sectional Area

Calculation is carried out in the pipe whose cross sectional areas of sub domains A, B, C are 1.0 : 0.8 : 1.5 respectively. Width of the pipe along the y-axis (vertical to the paper) of each sub domain is the same. The "main flows" in sub domains are $f(t) : \frac{1}{0.8}f(t) : \frac{1}{1.5}f(t)$ respectively, so that flow rates through the pipe at all cross sections in three sub domains become the same value. Results of calculation are shown in Figs.15-17. The same results as 3.2 are confirmed.

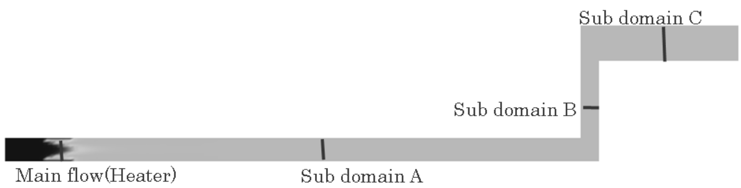
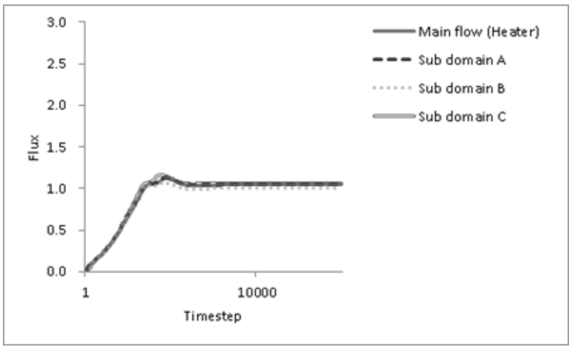
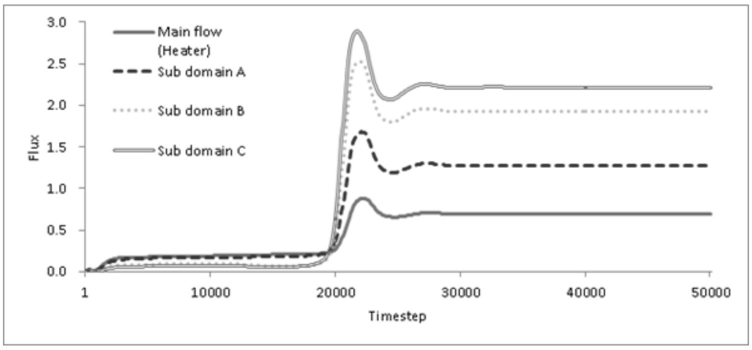


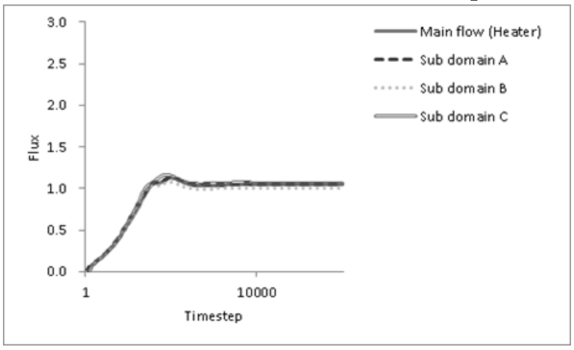
Fig. 14: Cross sections for calculation of flux.



(a) Proposed method.

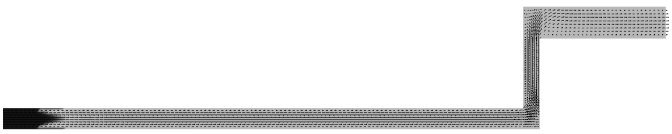


(b) Conventional method, same computational cost with (a).

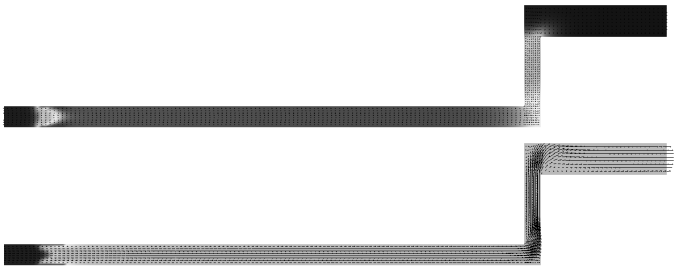


(c) Conventional method.

Fig. 15: Time history of variation of the flux of the pipe.

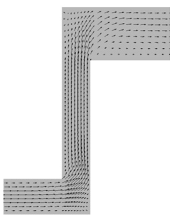


(a) Proposed method (Timestep=15000).

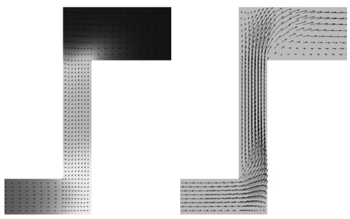


(b) Conventional method, same computational cost with (a) (Timestep=15000, 50000).

Fig. 16: Temperature and velocity vector of the pipe.



(a) Proposed method (Timestep=15000).



(b) Conventional method, same computational cost with (a) (Timestep=15000, 50000).

Fig. 17: Temperature and velocity vector near the joint region.

4 Conclusion

The method that is effective for the calculations of incompressible flow in a long region is proposed. In this study, 3-dimensional thermal convection in some orthogonally bended pipes is calculated using some sub domain grids.

The idea of the method is that the original flow is expressed as sum of the "main flow" and the variation from the main flow. The "main flow" is determined from the averaged velocity in the region surrounded by the heat source.

It is found that the steady flow of upward direction with high temperature is maintained, while it is not precisely by the calculation based on the conventional MAC method whose computational cost is the same range with the proposed method.

Acknowlegment

This work was supported by KAKENHI (Grant-in-Aid for Scientific Research(C): 22560157).

References

[1] F.H. Harlow and J.E. Welch, "Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface" *Phys. Fluids.* **8** (1965), pp.2182-2189.

[2] A.A. Amsden and F.H. Harlow, "The SMAC Method: A Numerical Technique for Calculating Incompressible Fluid Flows" *LOS Alamos Scientific Laboratory Report* LA-4370 (1970).

[3] T. Kawamura, F. Oono and Y. Nakamura, "A Numerical Method for Calculations of Flow in Long Region with Branch" *CFD Symposium, CFD2006* (2006), C3-5.

[4] T. Kawamura and K. Miyashita, "A numerical method for calculations of flow in long region with branch" *Natural Science Report, Ochanomizu University* **58** (2007), pp.45-55.

[5] T. Kawamura, "Numerical method for simulating thermal convection in a long channel standing vertically" *Natural Science Report, Ochanomizu University* **59** (2009), pp.37-46.

[6] A. Kuwana and T. Kawamura, "A Numerical Method for Thermal Convection in a Vertical Long Pipe", *Theoretical and Applied Mechanics Japan, Volume 60- 60th Japan National Congress for Theoretical and Applied Mechanics*, (2011), pp.155-162.

[7] K. Miyashita and T. Kawamura, "A Numerical Method for Flow in a Two-Dimensional Channel of Large Aspect Ratio" *Natural Science Report, Ochanomizu University*, **58**, (2008), pp.1-9.

Anna KUWANA
Center for Simulation Sciences, Ochanomizu University
Otsuka 2-1-1, Bunkyo-ku, Tokyo 112-8610, Japan
E-mail: kuwana.anna@ocha.ac.jp

Kanako IKEDA
Graduate School of Humanities and Sciences, Advanced Sciences, Computer Science, Ochanomizu University
E-mail: ikeda.kanako@is.ocha.ac.jp

Tetuya KAWAMURA
Graduate School of Humanities and Sciences, Ochanomizu University
E-mail: kawamura@is.ocha.ac.jp